COMPUTATIONAL STRESS AND MODAL ANALYSIS OF CAR CHASSIS

MOHAMAD TARMIZI BIN ARBAIN

A report submitted in partial fulfillment of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

NOVEMBER 2008

SUPERVISOR DECLARATION

We hereby declare that we have checked this project and in our opinion this project is satisfactory in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering with Automotive Engineering.

Signature :

Name of Supervisor:
Position:
Date:

Signature:

Name of Panel:
Position:
Date:

STUDENT'S DECLARATION

I hereby declare that the work in this thesis is my own except for quotations and summaries which have been duly acknowledged. The thesis has not been accepted for any degree and is not concurrently submitted for award of other degree.

Signature	:…………………………………
Name:	: MOHAMAD TARMIZI BIN ARBAIN
ID Number:	: MH 05013
Date:	: …………………………………

Dedicated to my beloved Parents,

ARBAIN BIN HAJI TUMIN,

SAEDAH BINTI DENAN,

Thank you for all the supports and encouragement during

This thesis is being made..

ACKNOWLEDGEMENT

Alhamdulillah,

I would like to thank my parents (Arbain bin Haji Tumin and Saedah Binti Denan), sister (Nursuliha), brother (Mohd Fitri and Mohd Adib Farid) for prodding, supporting, inspiring me to pursue higher education that eventually led me to fly across the country for pursuing this degree’s mechanical program.

I sincerely appreciate Mr. Mohd Shahrir B Mohd Sani for accept and giving me the opportunity to be my supervisor in order to finish this project. I am also grateful for his support and guidance that have helped me expand my horizons of thought and expression. Mr. Mohd Shahrir was very helpful in finding solutions to several problems I had during the finish this project. I am grateful to him for his time and patience.

I would like to thank Dr. Ahmad Syahrizan Bin Sulaiman and Dr. Daw Thet Thet Mon for providing me with the technical information required for this program. Special thanks not be forgotten should be given to my committee members. I would like to acknowledge their comments and suggestions, which was crucial for the successful completion of this project. All of their helps are very significant to the success of this project.

ABSTRACT

Chassis is one of the important parts that used in automotive industry and every car passenger has it. This structure was the bigger component in the car and the car shape dependent to this chassis. As a major component of a vehicle, chassis has a considerable affected to the performance of the car. Also known as the “back bone” of the vehicle, it will be subjected to mechanical shocks, and vibrations and the result were the failures some component and resonant was the worst problem can be happened. Therefore, the prediction of the dynamic properties of the chassis is great significance to determine the natural frequencies of the structure to make sure working frequency are lower than natural frequency of the chassis to avoid resonant and determine the stress distribution on the chassis when receive the load. The finite element modeling issues regarding the experimental analysis of car chassis is addressed for the natural frequency analysis (modal) by using FEMPRO Algor. A comparison of modal parameters from experiment and computational shows the validity of the proposed approach. Result shows that 1st bending for 1st natural frequency (50.56 Hz), 1st torsion for 2nd natural frequency (62.10 Hz), mixed for 3rd natural frequency (83.25 Hz) and 2nd bending for 4th natural frequency (91.89 Hz). The model performed the linear material stress analysis to define the stress distribution on the chassis when receive the load and the maximum stress of all cases are normally acting upon at the point of joint part but the value is under the allowable stress for steel which is 300 MPa.

ABSTRAK

Kerangka adalah salah satu bahagian penting yang digunakan di dalam industri automotif and setiap kereta pengangkutan mempunyainya. Struktur ini adalah komponen yang terbesar dalam kereta dan bentuk kereta bergantung kepada kerangka ini. Sebagai componen kereta yang utama, kerangka dianggap memberi kesan kepada prestasi kereta. Dikenali sebagai ‘tulang belakang’ kenderaan, ia akan menerima kejutan mekanikal dan getaran dan hasilnya adalah kegagalan sesetengah komponen dan resonan adalah masalah yang paling buruk yang akan terjadi. Sehubungan dengan itu, manganggar sifat dinamic kerangka adalah sesuatu yang bagus untuk mengetahui frikuensi asli kerangka untuk meghindarkan resonan dan menganggarkan taburan tekanan di dalam kerangka apabila menerima beban. Model unsur terkira (Finite Element Modeling) dibuat berpandukan kepada kajian eksperimen kerangka untuk kajian frekuensi asli (modal) dengan menggunakan FEMPRO Algor. Perbandingan modal parameter dari ekperimen dan pengiraan menujukkan kesahihan pendekatan yang dicadangkan. Keputusan menunjukkan bengkokkan pertama untuk frekuensi semulajadi yang pertama (50.56 Hz) , kilasan pertama untuk frekuensi yang kedua (62.10 Hz), campuran bengkokkan dan kilasan untuk frekuensi yang ketiga (83.25 Hz) dan bengkokkkan kedua untuk frekuensi yang keempat (91.89 Hz). Tekanan bahan mendatar (Linear Material Stress Analysis) untuk menjelaskan taburan tekanan pada kerangka semasa menerima beban dan tekanan maksimum untuk semua kes adalah biasanya bertindak pada titik persambungan dan tekanan yang di benarkan untuk besi waja adalah 300MPa.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF SYMBOLS

ω	Natural frequency
m	Mass
ε	Total strain, Bandwidth parameter
f	Frequency
t	Time
H (ω)	Frequency Response Function (FRF)
$\Delta\sigma$	Stress range

LIST OF ABBREVIATIONS

AISI	American Iron and Steel Institute
ASTM	American Society for Testing and Materials
CAD	Computer-aided design
CAE	Computer-aided engineering
DOF	Degree-of-freedom
FE	Finite element
FFT	Fast Fourier transform
FRF	Frequency response function
SAE	Society of Automotive Engineers

CHAPTER 1

INTRODUCTION

1.1 INTRODUCTION

Almost every year, each vehicle manufacture produce new design of their vehicles for them can compete to others manufacture. That means that the vehicles become important in nowadays lifestyle. The function of this vehicle was use to transfer or move people from one place to others places with safe and comfortable. These two important criteria implement in the every construction of the car. All these cars created commonly have to through the road that connected every place in the world and it created on the land with follow the earth land surface contour.

When Ford makes his first car, the car chassis was created from wood. After that, on about 1910's steel and aluminum was been use as the chassis of the automotive field effected by industrial revolution and the early of this year start use both woods and steel as the material of chassis. On 1930's created the technology that can improve the steel type and it come to improve the chassis structure in term of the increase the stiffness, torsion and reduction of vibration. This was the reason that the chassis was fully made from steel.

The chassis also receive the vibration and force that external and internal produce from the car. The road bumping, the load of passenger, the vibration of the engine and others can be the source of the external and internal force and it can be failure of the structure when the excitation of it coincides with the natural frequencies of the chassis and create resonant. Resonant held in two conditions first mode is bending and second twisting and it repeated with every natural frequency level [1]. Since the material almost same the different that differ for each chassis is come from the chassis design.

1.2 PROJECT BACKGROUND

Chassis is one of the major components of a vehicle because it can consider effected to the performance of a car. This can see when it be subjected to mechanical shocks or and vibrations that may result in failures some component and after some limit, it can also be major problem to the car such as the car can be collapse while it in running that cause from resonant. The resonant happen when the excitation same to the natural frequency of the chassis and important to determine the natural frequency of the structure to avoid this situation [2].

Finite element analysis is a computer simulation technique for modeling and analyzing the effect of mechanical loads and thermal stresses applied to a part or material that use in the system. This also tool to identifying the areas of stress concentration that are susceptible to the mechanical or thermal failure before manufacturing and test. It is the new method to define the parameter in save and short time because no waste sample material will produce and the result better and accurate. Validation of computational is important to make the result from both method is acceptable.

Through experimental method, it can define the properties of the structure using modal analysis [5].Therefore; the prediction of the dynamic properties of the chassis is of great significance. In this paper, the finite element analysis using 3D modeling issues regarding the experimental analysis of car chassis is addressed. The modeling approach is investigated extensively using both of computational and compared it to experimental modal analysis. A comparison of the modal parameters from both experiment and simulation shows the validity of the proposed approach. Then perform the computational stress analysis with linear material type analysis to find the stress concentration point in the car chassis. The point that come from the stress analysis can be use to determine the structure ability to withstand the load, force and the vibration.

1.3 PROBLEM STATEMENT

Growth economy gives affected nowadays lifestyle. Most people personally want drives theirs own car to work place. Every people know that car has a body which carries both the load and the weight. The car body consists of two parts: chassis and bodywork or superstructure. Seldom that car user realize that this chassis function to distribute the load and weight for whole body including the passenger to the suitable position in order to stabilize the car. Others function of this car chassis also have to withstand the vibration come from the mechanical part in the car and the vibration from the outer of car such as the road damping [7].

Steel was the material of the car chassis. It been used widely for chassis manufacturing among the car manufacture. The conventional chassis frame, which made of pressed members, can be considered structural as grillage. This chassis include cross-members located at critical stress point to provide that chassis structure box-like structure to absorb the impact from all angle. As the material that uses for chassis same, the different for every manufacture in their design to increase the performance of the car and this make each chassis design have their advantages.

This paper focuses to perform the finite element analysis to determine the stress and modal parameter of the car chassis and compared the result to the experimental data for validation. The model was follows the exact shape and dimension of the actual model. Finding of the stress points in the chassis is to analysis that it can withstand the load to provide the safety to the passengers of the car. Determination of the modal parameter important due the ensure that the working frequency of the car are lower than the natural frequency of the chassis to avoid the resonant [3].

1.4 PROJECT OBJECTIVE

There are several objectives regarding to the computational stress and modal analysis of car chassis which are:

a. Computational stress analysis of car chassis using FEA to determent the stress von misses distribution on the car chassis
b. Computational modal analysis of model of car chassis to determents the modal parameter such as the natural frequency and mode shape of the car chassis

1.5 PROJECT SCOPE

By starting this project based only on the objective is not recommended as is too large or too wide to cover, and it is important to create a scope of this project. Scope of Computational Stress and Modal Analysis of Car Chassis are:-

a) Design - create the 3D modeling of car chassis using CAD.

b) Analysis
   I. Linear material stress analysis of car chassis to find the stress von misses distribution.
   II. Linear natural frequency (modal analysis) of car chassis to find out the mode shape and natural frequency of the car chassis.

1.6 CHAPTER OUTLINE

Chapter 1 describes the purpose of Computational Stress and Modal Analysis of Car Chassis, the objective and the scopes of modal testing. This chapter also defines the problem and can be guide of the computational analysis.

Chapter 2 explains the fundamental of the Modal Testing and FEA information include the important the modal analysis to the chassis structure. It is important to study on the basic concept of modal testing for both side because both result use to verified the computational result.

Chapter 3 describes the procedure or the guided to archive the goal or the objective of this simulation. This chapter will explain the stage of the simulation where start from the design the model, detail of the procedure and tool to perform the simulation.

Chapter 4 provides the result of the simulation analysis and the discussion of every result. Comparison from experimental result to the simulation result is displayed and the result of convergence test. Selected the suitable meshing percentage can be find in this chapter.

Chapter 5 represent the summary of entire the simulation project include the recommendation for future research on the car chassis. This part relate the chapter 1 those the objective archive or opposite.

1.7 GANTT CHART

The purpose of Gantt chart is to display the time and duration together with work implementation. This reason Gantt chart created to ensure the progress in flow and it can be referred to Appendix A and Appendix B.

CHAPTER 2

LITERATURE STUDY

2.1 INTRODUCTION

This chapter explains the fundamental of the Modal Testing and FEA information that reason to determent the modal parameter of the chassis structure. It is important to study on the basic concept of modal testing for both side methods because both result use to verified the computational result.

2.2 MODAL ANALYSIS

Modal analysis is the study of the dynamic properties of structures when it under vibration excitation. Also known as Experimental Modal Analysis (EMA) is the field of measuring and analyzing the dynamic response of structures and or fluids when excited by an input [3]. The parameter to describe the structure in terms of its natural characteristics which are the frequency, damping and mode shapes. Modal domain becomes analysis domains to help to understand structural vibrations. Under normal operating conditions, a structure will vibrate in a complex combination of all its mode shapes.

By analyzing the mode shapes, it is possible to gain an understanding of the types of vibration that the structure can exhibit. Modal analysis also reduces a complex structure, which is not easily analyzed, into a set of single-degree-of-freedom systems that can easily be understood. In practice, a structure's natural frequencies cannot be defined until it is jolted, hit, or excited in some way [3]. As usual in physics, the system needs an input to get a response. Physical testing for normal modes excites the system and measures the response.

Easy to describe the modal analysis like freely supported flat plate which constant force is applied to the plate shown in the Figure 2.1 with input is from force and the output will measure at response sensor that attach to the plate.

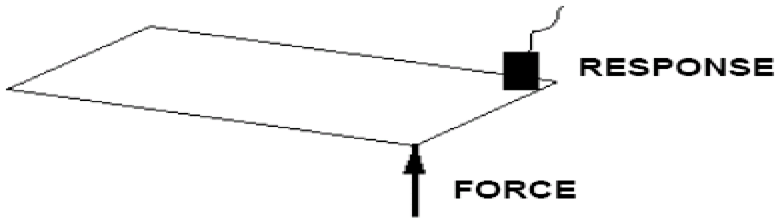

Figure 2.1 Simple plate and excitation/response model [4]

People will think when a force in a static sense which would cause some static deformation in the plate. But the fact is the force that applies to the plate varies in a sinusoidal fashion. Fixed force frequency of oscillation to make the constant force use as the input data.

There will change the rate of oscillation of the frequency but the peak force will always be the same value, only the rate of oscillation of the force will change. The response of the plate measured due to the excitation with an accelerometer attached to one comer of the plate, Figure 2.2, the result shown that the amplitude change as the rate of oscillation of the input is changed [4].

There will be increases as well as decreases in amplitude at different points as changed up in time. This result differs from expected since applying a constant force to the system then the amplitude varies depending on the rate of oscillation of the input force.

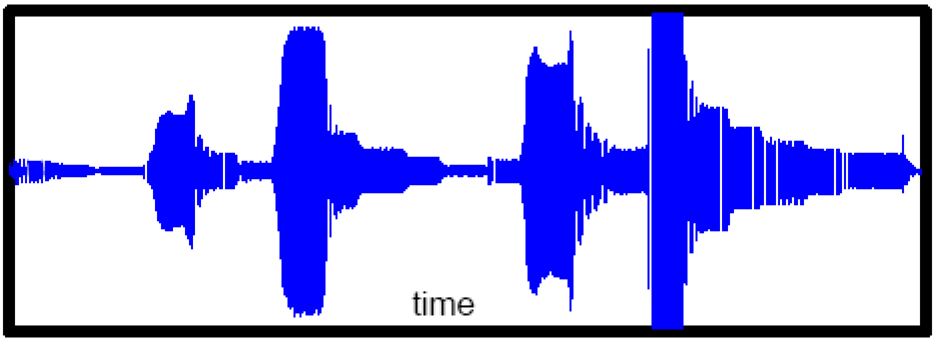

Figure 2.2 Simple plate responses [4]

Figure 2.2 indicate that the response amplifies as we apply a force with a rate of oscillation that sets closer and closer to the natural frequency or resonant frequency of the system and reaches a maximum when the rate of oscillation is at the resonant frequency of the system. This happen because the same peak force and just oscillation is changing.

Time data provides very useful information like in Figure 2.2, but in modal analysis using frequency domain is more useful due the calculation will depend on frequency value. Engineers use modal analysis to predict the theoretical vibration of a structure from a finite element model.

The first step is to represent the structure as a theoretical collection of springs and masses; then develop a set of matrix equations that describes the whole structure. Then apply a mathematical algorithm to the matrices to extract the mode shapes and resonant frequencies of the structure [1].

All this theoretical work produces very practical benefits because it allows the prediction of the modal response of a structure. Finding and addressing potential problems early in the design process, manufacturers can save money and improve product quality.

2.2.1 Frequency Response Function (FRF)

Time data provides very useful information for experimental result. Changing time data transform to the frequency domain using Fast Fourier Transform to get graph like Figure 2.3. The Frequency Response Function (FRF) is a fundamental measurement that isolates the inherent dynamic properties of a mechanical structure. Peak in Figure 2.3 function which occurs at the resonant frequency of the system where the time response was observed to have maximum response corresponding to the rate of the input excitation.

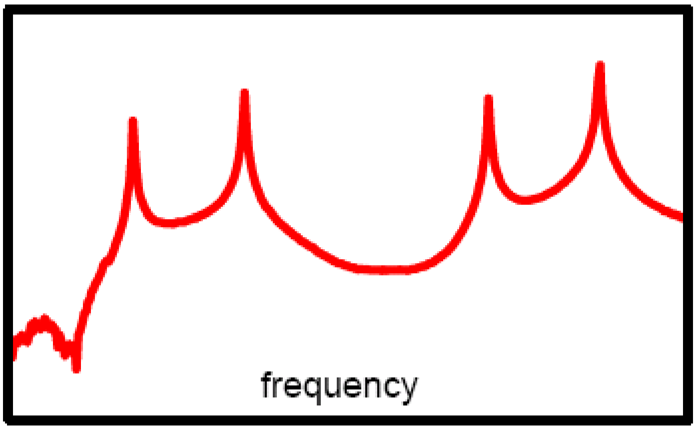

Figure 2.3 Simple plate frequency response functions [4]

Frequency domain can be use for to determine where the natural frequency occurs because peak of the frequency domain is the maximum amplitude which means the natural frequency value for system [4]. Clearly the frequency response function is easier to evaluate because this peak also the peak at time domain.

2.2.2 MEscopeVES (Visual Engineering Series)

This is a family of software packages and options that make it easier for to observe, analyze, and document noise & vibration problems in machinery and structures. ME’scopeVES is used to display and analyze experimental multi-channel time or frequency domain data, acquired during the operation of a machine, or forced vibration of a structure [3]. ME'scopeVES contains an interactive animated display